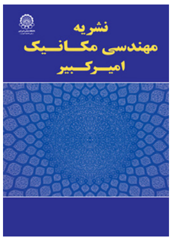

Amirkabir Journal of Mechanical Engineering

Amirkabir J. Mech. Eng., 51(2) (2019) 81-83
DOI: 10.22060/mej.2017.12784.5437

Effect of Support on Wind Flow Field Around Array of Two Inline Buildings

H. Haghighifard, M. Tavakol*

Department of Mechanical Engineering, Shiraz Branch, Islamic Azad University, Shiraz, Iran

ABSTRACT: In the present study turbulent wind flow field around inline surface-mounted and supported buildings has been investigated numerically. In order to model turbulence, re-normalization group *k-ε* and realizable *k-ε* turbulence models are employed. According to the numerical simulations, stream-wise velocity profiles around single surface-mounted and supported buildings are compared with the experimental data. Consequently, the validated model with realizable *k-ε* turbulence models is used to simulate flow field around two inline surface-mounted and supported buildings. Results have been reported for two Reynolds numbers (17000, 170000). Approximately, same velocity field was observed for non-supported buildings at two flow Reynolds numbers. Although, for supported buildings small difference is observed in the velocity profile under and above the building. Comparison of results for non-supported and supported buildings shows that behind the supported buildings the near ground reversed flow region was removed and lead to the lower drag force on such building. Moreover, supports increase the reattachment length on the upstream building.

Review History:

Received: 20 April 2017
Revised: 5 July 2017
Accepted: 16 July 2017
Available Online: 24 July 2017

Keywords:

Inline buildings
Supported building
Separation length
Reversed flow

1- Introduction

Flow field around buildings has a substantial influence on pressure distribution, stability of structure of a building, particle deposition and pollution. Determining flow around inline supported and non-supported buildings flow field is important to predict particle deposition pattern and air conditioning around these buildings.

In last decades flow field around various types of buildings and bluff bodies has been investigated experimentally [1,2]. Recently, governing equations for such flow field has been solved and presented numerically by means of commercial softwares. Among previous studies Castro and Robins [3] evaluated turbulent shear flow field around a cubic building model. In the study of Motalebi et al. [4], flow field around a *3D* non-supported building model and a *3D* supported building model has discussed by means of different turbulence models. Then proper turbulence model has been selected using comparison with experimental data. They showed that reattachment length behind supported building is shorter than that of for non-supported one.

In addition, in recent years experimental and numerical studies were conducted to evaluate flow field around model of inline buildings. Tulapurkara et al. [5] observed that the largest difference between velocity distribution for single and inline buildings occurred at their symmetry lines. In another studies, Mittal [6] and Gnatowska [7] investigated wind flow around two buildings with unequal height. They showed that flow structure around such buildings depends on various factors such as boundary layer thickness to building height, incoming flow turbulence intensity and distance between two buildings.

Based on the available literature, there is no research about interfering effect of wind between supported inline buildings. Therefore, in the present study air flow field characteristics around inline non-supported and supported buildings were investigated and compared two different spaces. All building model dimensions were scaled to 1/100 dimensions of the actual buildings. Dimensions of computational domain were selected according to guidelines of Architectural Institute of Japan (AIJ) [8]. These guidelines are very useful for simulating atmospheric boundary layer around buildings and creating a proper computational domain to solve the problem numerically.

In this study turbulent air flow around model of two inline building was evaluated numerically.

2- Methodology

In this study using a validated numerical model, velocity distribution around an array of two inline supported and non-supported buildings was presented and discussed. Numerical simulations were performed by commercial code FLUENT 6.3.26. In this regard, governing equations were solved by finite volume method and in order to solve coupled velocity and pressure field SIMPLE algorithm is applied. The convective terms were discretized using QUICK scheme and pressure terms were discretized with second order accurate central difference scheme. The converged solution is attained when continuity and momentum residuals reduced to 1×10^{-6}, Re-Normalization Group (RNG) *k-ε* and realizable *k-ε*

Corresponding author, E-mail: tavakol@iaushiraz.ac.ir

turbulence models were employed for turbulence modeling. Based on results, realizable *k-ε* turbulence model is used to simulate turbulent flow field around such building models.

3- Results and Discussion

According to validated model, realizable turbulence model is found to be more suitable for predicting the air flow field around the model of inline buildings. For comparison, results are presented for two different Reynolds numbers. Fig. 1 shows stream-wise velocity distribution at various sections around model of two inline wall-mounted buildings with distance *H* for two Reynolds numbers of 17000 and 170000. According to this Fig., it is clear that velocity distribution is same for two Reynolds numbers. Furthermore, reversed flow zones were observed over the upstream building.

Fig. 2 indicates stream-wise velocity distributions at various sections around two inline supported buildings. This Fig. shows that the near ground recirculation zone behind the obstacles was removed for supported models. In this case in subsurface section the pattern of flow was similar to internal channel flow and reversed flow was observed in the region between and behind the buildings. A comparison between results shows that for non-supported buildings change of Reynolds number from 17000 to 170000 has no significant effect on flow field, however, for supported buildings change of Reynolds number has a moderate effect on flow field, especially in the subsurface and over the buildings.

In Table 1, the length of separation bubble over the upstream building, X_{r1}, separation length in the windward side of the upstream building, X_{s1} and reattachment length behind the downstream building, X_{r3} are reported. Results show that length of recirculation zone above the upstream supported building is more than that of for non-supported one. Also, there is no recirculation zone downstream of supported buildings as seen in Fig. 2.

Table 1. Reattachment and separation lengths for model of two inline buildings, Re=17000

Type of building	X_{s1}/H	X_{r1}/H	X_{r3}/H
Non-supported (*S*=*H*)	1.67	2.67	3.33
Supported (*S*=*H*)	–	2.73	–

4- Conclusion

In this study, flow field around inline non-supported and supported buildings are compared by means of numerical simulations. In this manner, Realizable *k-ε* turbulence model is used. Results show that:

1. For flow Reynolds numbers of 17000 and 170000 air flow field was same for non-supported buildings. For supported buildings small difference was observed between flow fields especially behind the buildings.

2. The near ground recirculation zone behind the buildings was removed for supported buildings.

References

[1] Logan, E, and W. Schofield. "Turbulent shear flow over surface mounted obstacles." *Int J Heat Mass Transfer* 112 (1990): 376-385.

[2] Castro, I. "Measurements in shear layers separating from surface-mounted bluff bodies." *Journal of Wind Engineering and Industrial Aerodynamics* 7.3 (1981): 253-272.

[3] Castro, I, and A. Robins. "The flow around a surface-mounted cube in uniform and turbulent streams." *Journal of Fluid Mechanics* 79.2 (1977): 307-335.

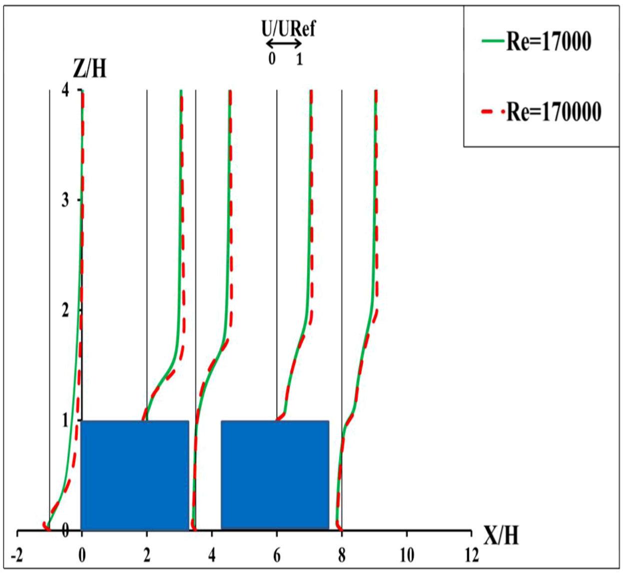

Fig. 1. Stream-wise velocity distribution in symmetry plane (*Y/H*=0) for two different Reynolds numbers around non-supported buildings for *S*=*H*

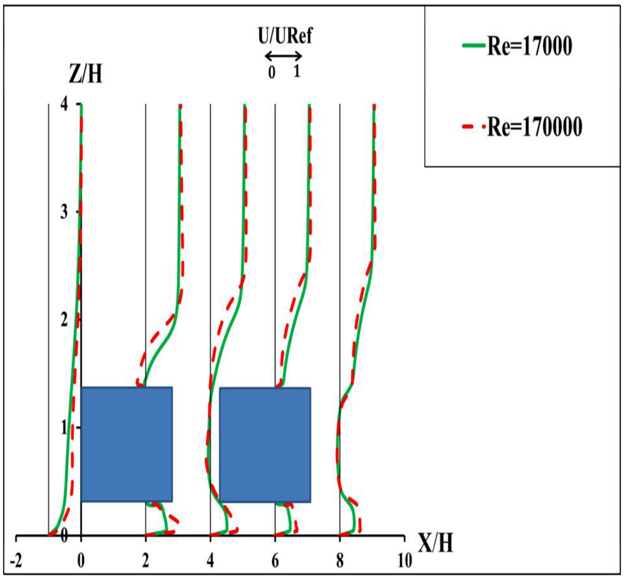

Fig. 2. Stream-wise velocity distribution in symmetry plane (*Y/H*=0) for two different Reynolds numbers around supported buildings for *S*=*H*

[4] Motalebi, S, Abouali, O, and Goshtasbi,E. " investigation of flow field around a building with subsurface flow" *Seventeenth International Conference on Mechanical Engineering*, 2009.

[5] Tulapurkara, E., Gowda, BHL, and Chaukar, JS. "Mean velocity field around prismatic bodies in tandem arrangement." *Journal of Wind Engineering and Industrial Aerodynamics* 93.10 (2005): 777-796.

[6] Mittal, A. "Wind flow simulation in the vicinity of tall buildings through CFD". *The 8th Asia-Pacific Conference on Wind Engineering*, 2013.

[7] Gnatowska, R. "Aerodynamic characteristics of three-dimensional surface-mounted objects in tandem arrangement." *International Journal of Turbo and Jet Engines* 28.1 (2011): 21-29.

[8] Tominaga, Y, Mochida, A, Yoshie, R, Kataoka, H, Nozu, T, Yoshikawa, M, and Shirasawa, T. "AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings." *Journal of Wind Engineering and Industrial Aerodynamics* 96.10 (2008): 1749-1761.

Please cite this article using:

H. Haghighifard, M. Tavakol, Effect of Support on Wind Flow Field Around Array of Two Inline Buildings, *Amirkabir J. Mech. Eng.*, 51(2) (2019) 81-83.

DOI: 10.22060/mej.2017.12784.5437